A4 Assignment – ESE516-SPRING 2019

DUE DATE: MONDAY FEBRUARY 25TH 2019 before 11:59pm EST (By almost midnight). To be submitted on Google Drive ON THE FOLDER OF YOUR TEAM!
Remember: Please submit an excel file of the **Complete** updated BOM generated by Altium. This BOM shall have all the components you need to buy for your project. Also submit a picture showing you completed a small Altium Layout Tutorial.

1.) Altium Bill of Materials [100 points]

For this assignment, now that you have all your components on an schematic, you will use Altium to create a full BOM that includes ALL your components (including resistors, capacitors, connectors, external components.).

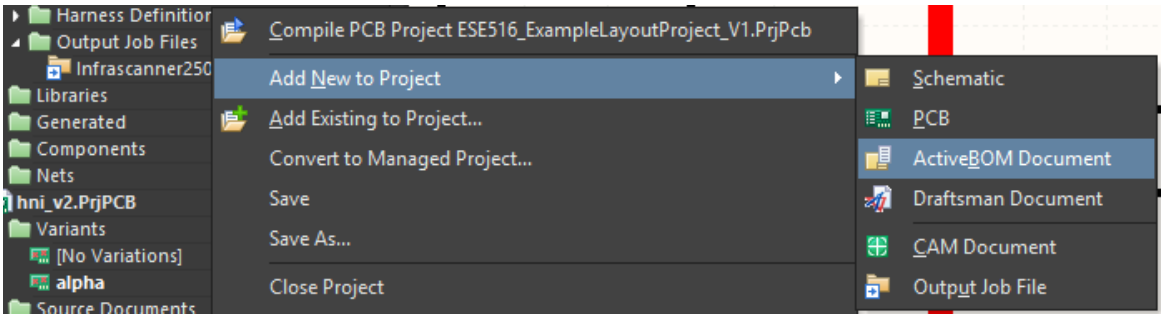
Steps:

1.) Familiarize yourself with the ActiveBom, a Bill of Materials file you can add to your project. Read:

https://www.altium.com/documentation/18.0/display/ADES/((BOM+Management+with+ActiveBOM))_AD

The previous document will explain what the document is and how to add a solution to a component (how to add a Manufacturer Part Number to a device). Please read it in its entirety and return to it if you have questions.

2.) Add an ActiveBom document to your project (right click on project and do Add New to Project -> ActiveBom Document). It might take a while to open.



3.) Once loaded, the ActiveBom documents will add a row for each of your components on the schematics. It is your task to add a solution to each component. The restrictions are:
   a. Components must be selected from Digikey. When you select components, please make sure to select components that are in stock and are the right footprint!
   b. If you have any component that is not available from Digikey, add the page where you can find it.
   c. If you are using external components, you need to add the external component (right click -> Add New -> Custom Item). REMEMBER to add price!
   d. If you are not mounting some components (like the flash memory)

4.) With your Teammate, please go through each item of the BOM and make sure the solution you added is the correct value, footprint, voltage tolerance, etc. Sometimes Altium can select a component with a different footprint if you let it select for itself.

   STUFF TO LOOK FOR
   a. Check that the footprint of the solution is the correct one
   b. Check that it is on stock in Digikey
   c. Check that the Minimum Order Quantity is 1 (don't put something where the MOQ is 10,000!)

d. Check that you chose the solution available on Digikey (check you chose the supplier is Digikey)

5.) Once you are done, on the toolbar go to “Reports ->Bill of Materials”. A new window will pup up where you can check what you are going to output. From this window, you can export an excel sheet.

6.) Open the excel sheet and add new rows at the top. Please put the following info:
   a. Team Name
   b. Team Members
   c. Please add a TOTAL column that adds the price of all the components on the BOM

What to Turn In:

In a folder called A4, please submit the excel sheet generated with this method. This excel sheet must:

- Have ALL the components from Digikey with valid solutions and price
  - Some sensors can come from somewhere else if not on Digikey- please add a note on those in another column
  - All resistors, capacitors, connectors, etc. Must be from Digikey
- Solutions must have a minimum order quantity of 1
- The excel sheet must have ALL EXTERNAL DEVICES included. Please check above on how to add additional rows to the ActiveBom project where you can add devices outside your PCB. Do not add wires/wire cost.
- Do not include SD Card cost
- Add any comments on another column if needed.

2.) Altium Layout Introduction [20 points][Individual]

For this part of the assignment we will follow Robert Fenarec[1]'s great simple Altium Tutorial Layout (available free on Youtube and Github).

Please download the PCB Altium project and follow along as Robert Fenarec explains how to lay down a board.

Tutorial Link: https://www.youtube.com/watch?v=2I2TX3RLEGM

GitHub Link (Download all the files):
https://github.com/robertferanec/board-led-youtube

The Github already has the PCBDoc added and routed. To follow the example, please right-click on the Project and do Add New to Project -> PCB. Then use this bare file to follow along.

This is to be done individually.

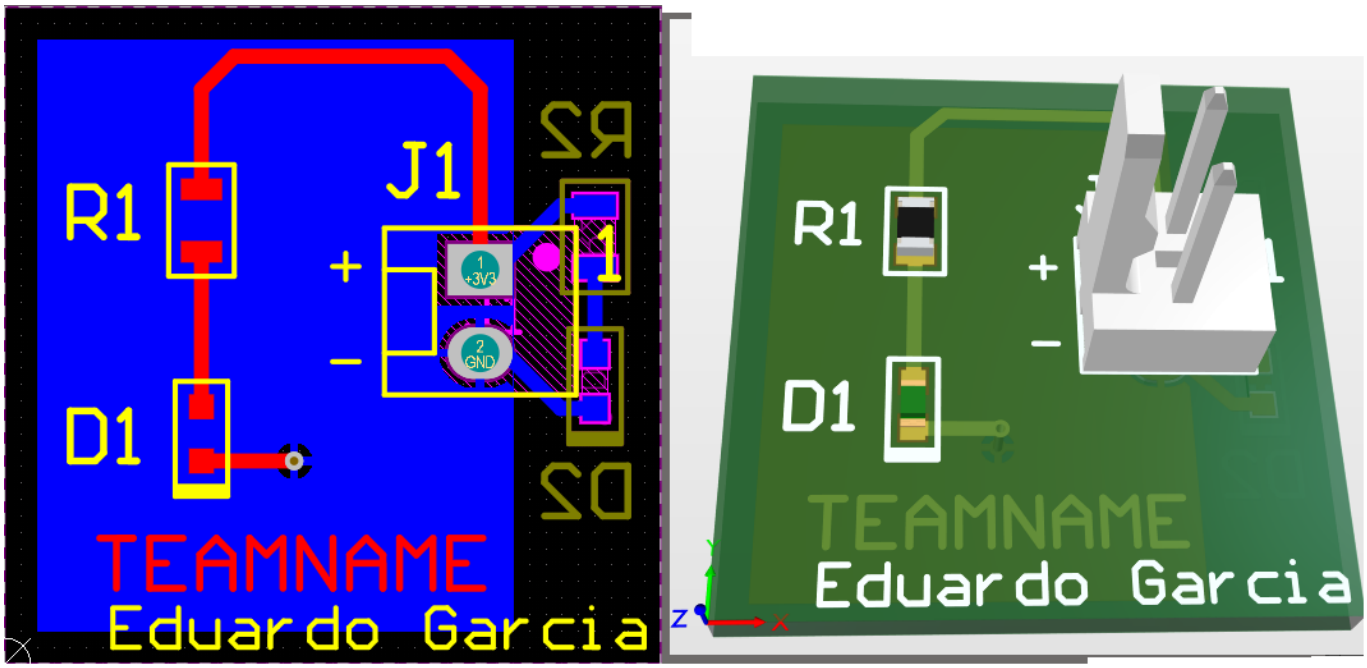
What to Turn In:

Please submit a picture of your board once done with the following changes:

- Add your group name on the top layer copper
- Add your name on the top layer silkscreen

Save a picture on the A4 folder with your name (example: ESE516_EduardoGarcia_A4.png) like the following:

[1] He runs a very nice class if you are interested in mastery in Altium: https://www.fedevel.com/academy/



Extra: There is a video continuation that build on this tutorial that is of great help. I deeply recommend following it! It will show you techniques that will make your life easier. No submission for this part – I trust you will take a look!

Video: https://www.youtube.com/watch?v=L36KicrU45Q